

Numerical Simulation of V-22 Aircraft Aerodynamics

Tsze C. Tai*

Naval Surface Warfare Center, Carderock Division
West Bethesda, MD 20817

Abstract

The aerodynamic flowfield about a wing-fuselage-nacelle configuration of the V-22 aircraft is simulated by using a multi-zone, thin-layer Navier-Stokes method. The work is primarily concerned with the forward-flight mode at a freestream Mach number ranging from 0.209 to 0.345 with corresponding Reynolds number ranging of 12.4 to 20.5 million based on the wing chord for flight at sea level. Effects of protuberances FLIR, AAR47 sensor fairing, and the refueling boom are also considered. Major flow features including the three-dimensional flow separation due to viscous-vortex interactions observed experimentally are captured by the simulation. The massive separation from the wing for flows at high angles of attack provide first approximations to the position of the wing vortex with respect to the tail surface location, a clue to the cause of the V-22 tail buffet problem. A brief description of the hover mode simulation is also given.

Introduction

The V-22 tiltrotor aircraft combines the high speed efficiency of a turboprop aircraft with the vertical takeoff and landing capabilities of a helicopter. Although the concept has been investigated for many years, it has only recently been developed into an operational prototype for military use.¹⁻⁵ The ability to take off and land anywhere and to fly like an aircraft has great potential for civil transport.⁶ The ongoing military program will provide an important database for civil applications.

The aerodynamics of the V-22 aircraft were primarily determined through experiments during its early stage of the development.^{1,2,5} Because of its vertical take-off requirement, the aircraft was designed “from inside out” and thus had design restrictions that adversely affect its aerodynamic performance. A comprehensive review on the V-22 Osprey aerodynamic development during the past 15 years is given by McVeigh et al.⁷ Recently, the use of the computational fluid dynamics (CFD) has been explored by the present author to analyze the complex aerodynamics of the V-22 aircraft. The overall flowfield about the aircraft in either hover or forward-flight mode has been simulated using a multi-zone, thin-layer Navier-Stokes method.⁸⁻¹² The aerodynamics in hover-to-transition flight mode of the aircraft was explored by Meakin based on a different CFD procedure.¹³

The present paper summarizes the work at Naval Surface Warfare Center-Carderock Division in simulation and analysis of V-22 aircraft aerodynamics during the past five years. The objectives are to develop a reliable tool for predicting the aerodynamic performance of the aircraft, as well as to investigate the source of drag rise and the tail buffeting at high angles of attack. As a prediction tool, the

CFD model must be detailed enough to include significant protuberances and cover the Mach range from subsonic to transonic speed (high subsonic freestream) in the design envelope.

* Senior Research Scientist, Marine and Aviation Department, Associate Fellow AIAA. Presented at the DoD High Performance Computing Users' Conference, Monterey, CA, June 7-10, 1999.

Simulation Method

The simulation method, or the computational method employed includes two main elements: the grid generation and the flow solving. The description of the method therefore will be centered around these two topics which are of equal importance.

Geometric Configuration and Grid Generation

A structured, curvilinear, body-conforming grid is employed. The surface grid is constructed from the manufacturer's blueprint with refinement based on data provided by Boeing Defense & Space Group. A cylindrical grid topology, which is basically made up of an H-O mixed type with H-type in the longitudinal plane and O-type in the crossflow plane, is adopted. The outer cylindrical surface is set at 7.7 chord lengths from the aircraft centerline. For the "clean aircraft" configuration in which symmetry about the centerplane can be assumed, only the right half of the aircraft needs to be modeled. For configuration with the refueling boom which is located on the right nose, the whole aircraft must be modelled. The latest model consists of the wing, fuselage, nacelle, forward-looking infrared, AAR47 sensor, and the refueling boom. The tail sections are now being added as well as the wing vortex generators and the strakes on the forward portion of the fuselage.

The advantage of this grid topology is that it permits simple and accurate discretizing approximations, and can be easily clustered in the viscous region. It is relatively fast for flow solving with good accuracy. Its primary disadvantage, however, is that the modeling is difficult and time-consuming for complex geometries.

In generating the grid, we first divide the whole domain into upper and lower regions. The multi-block scheme is applied to generate the common interface mesh which is then used as the base boundary in generating the basic grid for each region. The two meshes are subsequently combined together into a single basic grid. For the basic wing-fuselage-nacelle configuration, the geometry is relatively more complex than other aircraft because of the nacelle mounted at the wing tip and the sponson protrusion at the bottom of the fuselage. To overcome the difficulty, a new successive multi-block grid generation procedure was developed. A brief description on the procedure is given in Ref. 9. The final grid, after viscous clustering, has a total of 179 x 105 x 57 points covering the longitudinal, circumferential, and radial directions, respectively.

The NASA Ames 3DGRAPE code¹⁴ is used for basic grid generation and CNS/ZONER code¹⁵ for zoning and clustering. The latter has been modified to cluster the viscous region with a variety of spline curves for the radial direction to be fitted.

External Components -- Protuberances

The external components (protuberances) to be considered here include the forward-looking infrared unit (FLIR), the fairing of the electronic sensor (AAR47), and the refueling boom. They are described below along with the resulting grid changes due to these components.

a. Forward-Looking Infrared (FLIR)

The forward-looking infrared unit (FLIR) is a short hemi-spherical cylinder mounted upside-down directly beneath the aircraft nose. The cylinder has a diameter of about 10.5 inches and a mean height of 12.5 inches. Grid resolution in the local area has been enhanced in both longitudinal and circumferential directions to accommodate for the increased complexity caused by addition of the unit.

The resulting volume grid has a total of $185 \times 110 \times 57$ points covering the longitudinal, circumferential, and radial directions, respectively. Since the unit is mounted with its centerline coincides with the aircraft centerplane, the symmetry assumption remains valid and therefore only the right half of the aircraft needs to be meddled.

b. AAR47 Sensor

The fairing of the AAR47 sensor is a raised convex surface geometry with a height of approximately 3.0 inches. Shaped like the back of a turtle, the unit is mounted on the side of the aircraft nose below the front of the wind-shield panel, one on each side. Thus, the assumption of symmetry still holds. The size of the volume grid is increased to $203 \times 115 \times 57$ when the AAR47 fairing is implemented. The components are embedded seamlessly in the main surface with no increase in complexity in flow solving. The geometry of the AAR47 sensor fairing has been modified by Boeing after the present work. The new shape has an elongated rear portion to reduce the pressure drag due to the component.

c. Refueling Boom

Unlike the FLIR and AAR47 units, the refueling boom is installed on the right side of the nose only. It is a 69-inch long blunted ogive-cylindrical tube having a diameter of 4.5 inches. The boom is mounted between the FLIR and AAR47 units. Because of the one-sided location of the boom, the assumption of symmetry is no longer valid and therefore the whole aircraft must be modelled. The size of the volume grid is increased to $215 \times 231 \times 57$, which is more than double the size of the previous grid. Because of increased complexity, the nose portion of the overall grid is completely revised in order to incorporate the boom. Again, the gridding is accomplished in a seamless manner. Likewise, the number of zones for the flow solving is more than doubled.

Flow Solver and Turbulence Model

The NASA Langley thin-layer Navier-Stokes code, namely the CFL3D code¹⁶ with multi-zone capability, is used as the basic flow solver. Appropriate modifications to the code for applying specific boundary conditions are implemented. The code is based on a finite volume algorithm with a spatially-factored diagonalised, implicit scheme. The upwind-biased-differencing technique is used for the inviscid terms and central differencing for all viscous terms. The method is globally second-order accurate and well suited for patched grids in a multi-zone domain.

The code is upgraded with a variety of turbulence models, including the basic Baldwin-Lomax algebraic model¹⁷, the Spalart-Allmaras one-equation model¹⁸, among others. These turbulence models have been carefully examined and evaluated by Rumsey and Vatsa¹⁹. In the present work, the B-L algebraic model with a Degani-Schiff type modification²⁰ along with the Tai extension¹¹ is used for cases with small-to-moderate angles of attack. The B-L model is used widely throughout the computational fluid dynamics community. Although simple, it is known to be the best model for flowfield dominated by vortical flows.

The Baldwin-Lomax model was extended to three-dimensional flows by the present author¹¹ to remove some abnormalities associated with a curvilinear grid for complex geometries. In the modification, the directed normal distances (the “viscous” coordinates) of the original model (with or without the Degani-Schiff type modification) is replaced by curvilinear radial coordinates in a general curvilinear coordinate system. The extension, which makes the model more compatible with the Navier-Stokes equations solved in curvilinear meshes for a complex geometry, offers physically realistic “viscous” coordinates in case of highly curvy grid lines. The present V-22 grid is a perfect example having highly curvy grid lines. The new model yields slightly higher eddy viscosity and thus the skin friction values than the original model. Any grid that works well with the original model will also work well or better with the modified model.

For flow at high angles of attack, the B-L model starts to cause convergence problem because of the unsteadiness due to large flow separation, thus the Spalart-Allmaras turbulence model is employed as a back-up solution.

Multi-Zone Technique and Boundary Conditions

The single basic grid generated above is eventually divided into multi-zones for flow solving using the multi-zone technique.²¹ In the literature, the terms “multi-zone” and “multi-block” are generally interchangeable because a particular “block” of the grid generated is used also as a “zone” in the flow solving. In the present paper, however, we will distinguish the “multi-zone” from “multi-block” because they are NOT the same. In the case of the wing-fuselage-nacelle configuration, we have divided the single basic grid into 21 zones for flow solving, while the grid was generated with 25 blocks.

The division of the overall mesh into multi-zones depends primarily on the convenience in applying the boundary conditions. Having the same number of radial mesh points for both viscous and inviscid zones, such a division offers exact coincident boundaries between the zones. The main advantage of the approach is that the conservation of the spatial flux of mass, momentum, and energy between the zones

is automatically satisfied. A total of 44 interface boundaries are set among the 21 zones in flow solving.

The boundary conditions for the Navier-Stokes flow solver are, within a particular zone: (1) Freestream condition imposed at upstream (for the most forward zones only), (2) Freestream pressure recovery in downstream (for the most rearward zones only), (3) Characteristic form of inflow-outflow at the cylindrical outer boundary, and (4) Viscous nonslip flow at all solid surfaces (wing, fuselage, and nacelle). The inlet and outlet of the nacelle are closed for simplicity, and the disc loading at the rotor is neglected. The reason for neglecting the disc loading is that there are only 5-6 ft/sec velocity differentials at the rotor disc which may have little effect on the resulting flowfield.

Results of Wing-Fuselage-Nacelle Configuration

For the wing-fuselage-nacelle configuration, simulated results are obtained for freestream Mach numbers between 0.209 and 0.45 at angles of attack of 7, 12, 16 and 18 degrees. These conditions yield a Reynolds number range from 12.4 to 26.7 million based on the wing chord length of 8.33 feet. Flow conditions corresponds to the wind-tunnel test condition at Boeing were also considered in earlier calculations with a Reynolds number less than 2.0 million based on the 15 percent scale model. Because of the large Reynolds number involved, all computations were performed by assuming a fully turbulent flow so there is no need to specify the transition location. The results are presented in the form of particle trace, velocity vector plots, and pressure distributions over the configuration. The case of angle of attack of 16 degrees will be discussed in detail because of important physical features involving separated flow and its direct impact on tail buffeting.

Converged steady-state results were obtained in about 4,500-5,500 iterations requiring approximately 10-12 hours of Cray C-90 CPU time for the half-aircraft model. The CPU time becomes doubled with the refueling boom added. Convergence is reached the solution residual drops 3-4 order of magnitude and the lift and drag coefficients asymptotically approached to constant values. For cases at high angles of attack with the modified Baldwin-Lomax model, the results were considered to be "converged" when the lift and drag coefficients fluctuate only within a narrow band. The fluctuation is believed to be caused by shifting of vortex cores created by massive crossflow (vortex-type) separation. Earlier computations were performed on the NAS (Numerical Aerodynamic Simulation) supercomputer facility at NASA Ames Research Center before the DoD High Performance Computing facilities became available in early 1996. Since then the author has used the facilities at NAVO as well as CEWES and WPAFB.

Particle Trace

The particle traces of the streamlines emanating from various stations at the fuselage, wing and nacelle for flow at angles of attack of 7, 12, and 16 degrees were plotted and will be presented at the oral version of the paper. The flow over the wing is attached for the case of angle of attack at 7 degrees. However, instantaneous streamlines originating at the sponson rolled up in the rear region creating free vortices that converge into the main stream downstream.

As the angle of attack increases to 12 degrees, the flow on the fuselage and most of the wing remains mostly attached. Streamlines over the top of the fuselage (overwing fairing) start to separate and roll up downstream, merging with those from the sponson. These separations are mainly due to the convergence of viscous streamlines inside the boundary layer. The phenomenon is known as viscous-inviscid interaction.

Further increase in the angle of attack to 16 degrees causes massive separation to take place over the upper wing surface, near the wing-nacelle juncture and from the mid-wing to the wing root. Unlike aerodynamic flows in most aircraft, the crossflow over the wing flows inboard towards the fuselage. The reason for this unusual crossflow direction seems to be due to the six-degree forward swept angle of the V-22 wing. A rear view of the particle traces of the flow gives a rather clear picture of the whereabouts of the free vortices resulting from flow separations over the wing, the fuselage, and the sponson. These vortices tend to merge into a single main vortex passing by the rear fuselage and finally the tail section. (The tail section is not included in the computation). The swirling of the flow after the wing-fuselage juncture has increased significantly by the large amount of crossflows and therefore a vortex breakdown may occur before the flow proceeds downstream. A similar flow pattern containing a core of separated flow was observed experimentally by McVeigh et al.²²

Velocity Vector

Velocity vector plots over the wing-fuselage-nacelle configuration at aforementioned angle-of- attack range are presented. At angle of attack of 7 degrees, the flow is attached; all the velocity vectors at the wing root point in the streamwise direction without any flow reversal. The attached flow is sustained to an angle of attack of 12 degrees from which a similar flow pattern is found.

This situation changes in the case of the 16-degree angle of attack. Velocity profiles and the corresponding crossflow components at the wing mid-span and the wing root (overwing fairing) are examined. At the wing mid-span, the flow approaches a zero velocity at the surface about two-third chord length from the leading edge, and starts to reverse thereafter. The streamwise velocities form a vortex right above the trailing edge. The separation resembles that of the familiar two-dimensional boundary layer; but it belongs to the three-dimensional type because of the presence of crossflow components. The crossflow changes its direction from outboard in the fore region of the wing to inboard in the rear. Because of massive separation (vortex-layer type separation as explained below), the crossflow velocities have increased sharply aft of the trailing edge.

Generally, the flow separation over the wing contains two types of separation: the bubble type and the vortex-layer type. The bubble type separation is due to vanishing skin friction, while the vortex-layer type is caused by convergence of streamlines in the spanwise (crossflow) direction. Between the two, the vortex-layer type separation usually prevails on the wing surface.^{24,25} One symptom for the prevalence is that the line of crossflow reversal precedes the line of zero-skin-friction. The location of the zero-skin-friction moves from two-third chord at the wing mid-span to mid chord at the wing root. On the other hand, the change of crossflow direction takes place from two-third chord at the wing mid-span to one-third chord at the wing root. The flow is therefore dominated by the crossflow effect more

than by the viscous effect. As a result, free vortices are created and proceed downstream leaning toward the inboard spanwisely. Before reaching the plane of symmetry, the crossflow turns into the streamwise direction, flourishing the swirling rate of the flow behind the wing-fuselage juncture. The above velocity profiles confirm the presence of large crossflow over the wing that creates strong vortex-type separation as discussed in the previous section.

Pressure Distribution

The surface pressure distribution over the wing-fuselage-nacelle configuration for the case of an angle of attack of 16 degrees is discussed. On the wing surface, peaky pressures appear in the leading edge region, and a large positive pressure pocket in the inner part of the wing results in from massive separation. In three-dimensional flows, the surface pressure influences the boundary-layer behavior in two ways: (1) the boundary layer being acted upon by the pressure distribution, i.e., the usual boundary layer development; and (2) the viscous streamlines inside the boundary layer being acted upon by the external streamline pattern, which is dictated by the pressure gradients. The latter is intrinsic to three-dimensional flows and becomes more dominant when the vehicle is at an high angle of attack. The pressure pattern on the wing surface confirms the dominating vortex-layer type separation in the inner rear portion of the wing.

Cross-sectional pressure distributions at three longitudinal locations, truncated at the zonal boundary for the nacelle, are considered to confirm the results of particle traces and velocity vector plots. Upstream of the sponson, which is also ahead of the wing, the pressure variation is smooth; pressure coefficients take on positive values in the lower region and start to decrease gradually above the “interface” plane. A high pressure coefficient of 0.44, apparently caused by the presence of the sponson, is spotted at the lower left corner of the fuselage. In the fore wing section, which is located just before the shoulder of sponson, the positive and negative pressure coefficients are clearly partitioned by the wing. The lower-most pressures lie on the surface indicating the flow thus far is attached. However, some crossflow activities start to take place near the wing-nacelle juncture and the wing root. As we proceed downstream, the crossflow activity magnifies and massive flow separation becomes apparent as the low pressure pockets develop above the wing surface.

Tail Buffet Investigation

The flow patterns revealed by particle traces aid investigation to the tail buffet problem of the V-22 aircraft. As discussed above, the wing vortex generated by the massive flow separation over the wing create strong flow swirling after it mixes with that from the fuselage, causing a vortex breakdown before the flow proceeds to the wake downstream. Vortex breakdown may occur during transient high-angle-of-attack maneuvers even when no breakdown is observed in steady freestream at the same angle of attack.²³ Previous flight tests indicate the V-22 aircraft suffers a tail buffeting during flight at high angles of attack. The results of particle traces provide first approximations to the position of the wing vortex with respect to the tail surface location. This result is supported by further analyzing the vorticity field in the region between the wing and tail sections. The concentrated vortex cores immediately behind the wing divide into several small cores as it proceeds downstream. Because of the flow high swirling rate,

a vortex breakdown is very likely before it reaches the tail section. Although the phenomenon of the vortex breakdown is not considered in the present steady-state solution, it nevertheless provides a clue to the cause of the tail buffet problem without considering the unsteady interaction of the wing vortices with the tail surfaces. This finding is significant in that it is for the first time (April-May 1994) to pin down the cause of the V-22 tail buffeting. The result was confirmed three months later by Boeing's wind-tunnel tests.²²

Drag Analysis

The results from the particle traces, velocity profiles, and pressure distributions can all aid to analyzing the aerodynamic drag of the aircraft. Basically the aerodynamic drag consists of the form drag and the skin friction. The form drag for V-22 is mainly the pressure drag plus a small amount of induced drag. Because of surface bluntness and spontaneous occurrence of three-dimensional flow separation, the V-22 has very high ratio of the pressure drag to the skin friction drag. The ratio is about three times higher than that of a commercial aircraft. This suggests that there is a good potential for drag reduction.

Effect of Transonic Flow

The CFD simulation allows exploring the V-22 aircraft to high Mach number flight conditions which could be difficult and risky in actual flight tests. Simulated flowfields containing embedded supersonic pocket are obtained at freestream Mach numbers between 0.41 to 0.45. The onset of transonic flow depends on the combination of Mach number and angle of attack. A local supersonic flow can appear at a freestream Mach number as low as 0.41 at a moderate-to-high angle of attack. The early onset of transonic flow for V-22 is due to the relatively high thickness ratio (23 percent) of the wing airfoil section. Transonic flow has adverse effects on both aerodynamic drag and tail buffeting. The aerodynamic drag increases because of added wave drag and possible shock-induced flow separation; while the tail buffeting may be worsened by increased flow separation. Also, effort to alleviate tail buffeting by vortex control device may be weakened by the supersonic pocket.

Comparison of Turbulence Models

Results discussed were obtained based on the Baldwin-Lomax turbulence model with the Degani-Schiff type modification plus the Tai extension. As mentioned earlier, however, some convergence problems were encountered in cases of angle of attack of 12 and 16 degrees. While these results are considered to be qualitatively correct, the case of the 16-degree angle of attack was also solved by using the Spalart-Allmaras one-equation turbulence model. Particle traces of the converged solution based on these two models are compared. The S-A model yields more extensive viscous-inviscid interactions near the nacelle-wing juncture but smaller crossflow velocities in the rear of the wing. The B-L model, on the other hand, reveals strong vortical flows in the rear wing region as observed in the wind tunnel measurements.

Effect of External Components

Steady-state results are obtained for the following cases: (1) the wing-fuselage-nacelle configuration with the FLIR added, (2) the configuration with both the FLIR and AAR47 fairing added, and (3) the configuration with the FLIR, AAR47, and the refueling boom added. The freestream Mach number is set at 0.209 and 0.345 with the angle of attack varying at 0, 7, and 16 degrees. The Mach 0.209 freestream speed was chosen to match the flow conditions used in wind-tunnel tests of a 15% scale model at Boeing, while Mach 0.345 is more in line with flight tests. Again the flow Reynolds numbers fall well within the fully turbulent flow range so a fully turbulent flow can be assumed.

Effect of FLIR

For the case of Mach 0.209 at zero angle of attack, the overall pressure distributions for both the wing-fuselage-nacelle configuration and the configuration with FLIR are almost identical except in the local area where the FLIR is installed. The FLIR's front face has high pressures while its rear side is subject to low pressure, resulting in a slight increase in drag coefficient of about 1.5 percent comparing with the clean configuration. This amount of drag increase, which could be solely due to the added component itself, agrees well with the manufacturer's estimate.

The effect of FLIR becomes more visible with the aid of resulting flow patterns given by particle traces. At an angle of attack of seven (7) degrees (cruise flight), the particle traces reveal that the flow patterns are similar for both configurations, except that streamlines over the upper rear fuselage of the configuration with FLIR tend to lie closer to the surface. This phenomenon could be caused by vortices generated due to the presence of FLIR which incidentally serves as a vortex control device. It indeed enhances the lift by a small amount as we compare the lift values of both configurations.

This favorable effect (increased lift) quickly disappears as the angle of attack increases. At an angle of attack of 16 degrees (maneuvering flight), flow separation over the upper wing surface becomes massive and large crossflows flowing in-board are apparent. The flow separation appears to be worsened by the presence of the FLIR geometry as evidenced by extensive flow swirling over the overwing fairing compared with that of the clean configuration. Since flow separation over the wing has a direct bearing on tail buffeting, the FLIR therefore has an adverse effect on tail buffeting at high angles of attack. That is, the onset of a tail buffet may occur at an angle of attack one or two degrees sooner with the FLIR installed. Consequently, there is also a small reduction in the lift coefficient.

Effect of FLIR and AAR47

Unlike the FLIR ball, addition of the AAR47 fairing has very little effect on the drag at zero angle of attack. This is because the unit is relatively "thin" comparing with the FLIR and appears to be more streamlined. The added drag may well be compensated by the favorable effect due to the free vortices created. The net lift-to-drag ratio is unchanged at zero angle of attack.

At a moderate angle of attack of seven (7) degrees, more vortices are generated that enhance the favorable effect on lift and drag. The particle traces of the flow reveal that the flow pattern resembles that of the configuration with FLIR alone. The streamlines over the fuselage remain close to the surface

which is beneficial for lift: An increase of 0.5 percent in lift is obtained as compared with the clean configuration. Further, the streamlines from AAR47 pass through the middle portion of the lower fuselage-sponson area where some flow separation might have been suppressed or eliminated by the vortices due to AAR47. A 1.2 percent decrease in drag coefficient is achieved as compared with the clean configuration. It results in a slight increase in the lift-to-drag ratio. Although small in magnitude, this is meaningful in that there are no penalties in adding these external components in cruise flight. The results of particle traces are also supported by velocity and pressure plots.

As the angle of attack increases, these benefits gradually disappear because of worsened flow separation. At 16-degree angle of attack, lift coefficient has decreased to about one percent below the clean configuration level, while the drag remains the same, resulting a net reduction in lift-to drag ratio. The increased flow separation also leads to a worsened tail buffeting

Effect of Refueling Boom

The effect of the refueling boom is evaluated with both the FLIR and AAR47 units on. In evaluation, therefore, we have to make comparisons of the results with that of the configuration with the FLIR and AAR47 geometries on. One obvious effect of the boom is that the flow is no longer symmetric with respect to the aircraft centerplane. The resulting lift coefficient is slightly higher on the left-hand side than that on the right, even there is no side slip. Although the direct viscous drag due to the boom itself is insignificant, the total drag is increased slightly. That is, the effect of the boom at this moderate angle of attack is unfavorable as opposed to the cases with the FLIR and AAR47 units.

As the angle of attack increases to 16 degrees, flow separation worsens as in the previous cases, accompanied by slightly unsymmetric flow distributions. Again, it has direct impact on the tail buffeting.

Because of unevenness of the flow, the vertical tail on the left-hand side would likely experience buffeting slightly more than the one on the right. By the same token, the left wing has a slightly higher lift than the right one.

Hover Mode Simulation

The same multi-zone thin-layer Navier-Stokes method was used to simulate the flowfield of a V-22 aircraft configuration hovering over a ground. Two hover heights, 14 and 25 feet, were considered. The numerical model consists of the wing, the fuselage, and the rotor-nacelle assembly tilted at 90 degrees with respect to the wing. The simulated flowfields contain realistic outflow characteristics and showed fairly good agreement with flight test data.

Grid Topology for Hover Mode

A special grid topology is developed to accommodate the physical flow. The topology is basically of the O-type in the crossflow plane having centerline that coincides with the centerline of the rotor-nacelle assembly. The shape of the overall grid resembles a Mexican hat. The grid consists of a suction section on the top of the “hat,” a disc loading zone, a wing-fuselage section, a flow mixing region, and a diffusion zone. Symmetry about the centerplane is assumed so that only half of the aircraft needs to be modeled. The complete grid for the half model has a total of 97 x 63 x 50 points generated with five blocks.

Flow Solving and Boundary Conditions

The same CFL3D code was used as the flow solver. The boundary conditions for flow solving are: (1) Specified disc loading at the rotor, (2) Induced flow at upstream, (3) Atmospheric pressure recovery at downstream, and (4) Viscous nonslip flow at all solid surfaces (wing, fuselage, nacelle, and ground). The inlet and outlet of the nacelle are closed for simplicity. For B.C. 1, the disc loading distribution is determined by the CAMRAD/JA code²⁶ for a given rotor power and the aircraft take-off gross weight. Alternatively, a constant mean flow velocity estimated by a simple relationship between the velocity, rotor power and aircraft weight can be also used at the rotor disc. For B.C. 2, the concept of “induced flow” is introduced and to be imposed as the freestream because the vehicle has no freestream velocity under a steady-state flow assumption. The magnitude of the induced flow velocity is determined by the amount of flow entrained at the rotor.

Results of Aircraft 25-Feet Above Ground

The case of the V-22 hovering at 25 feet above the ground is discussed. The disc loading is determined based on a rotor power of 3,670 hps and an aircraft gross weight of 42,000 lbs. An entrainment of 4.0 is used in determining the induced velocity for the upstream boundary condition.

Numerical results show that the flow is locally separated in the regions in front and behind the aircraft, but remains mostly attached in other areas. The velocity contour indicates high velocities in the front and rear regions, but moderate in most circumferential areas. Zones having the highest velocities are located somewhere between 30 to 100 feet from the center of the rotor in the front and rear of the aircraft, more so for the front region than the rear. The height of high velocity pocket varies from 1 to 10 feet above the ground. Velocity profiles at two azimuth locations, $\beta = 0$ and 30 degrees ($\beta = 0$ corresponds to the centerplane in front of the aircraft), are compared with the mean velocities measured by Meyerhoff and Gorge.²⁷ The agreement between the simulation results and the measured mean velocity profiles is fairly good. The velocity magnitudes are higher and profiles fuller in the $\beta = 0$ deg plane than those in $\beta = 30$ deg plane. In $\beta = 0$ deg plane, the velocity profiles at 105 feet radius are significantly fuller than those at 57 feet radius. The reason for this may be due to lack of sufficient length for development of the turbulent shear layer involved.

Conclusions

The V-22 aircraft aerodynamics in both forward-flight mode and hover mode are simulated by using a multi-zone, thin-layer Navier-Stokes method. Based on results obtained, which have been assessed with the wind tunnel measurements as well as flight test data, some conclusions may be drawn:

1. At an high angle of attack, the vortex-layer type separation resulting from the upper wing surface, the sponson, and the fore fuselage are strong to create free vortices that pass by the rear fuselage region, including the tail section. The massive separation from the wing for flows at high angles of attack provide first approximations to the position of the wing vortex with respect to the tail surface location, a clue to the cause of the V-22 tail buffeting.

2. The surface bluntness and spontaneous occurrence of three-dimensional flow separation are responsible for the aircraft's high ratio of the pressure drag to the drag due to skin friction. There is a good potential for drag reduction.

3. The high thickness ratio wing leads to early onset of transonic flow which has adverse effect on both drag rise and tail buffeting.

4. Effect of external components FLIR and AAR47 on the forward-flight aerodynamic performance is favorable at a moderate angle of attack due to favorable free vortices created. As the angle of attack increases, the effect becomes unfavorable due to worsened flow separation that leads to earlier tail buffet onset and slightly lower lift-to-drag ratio. The effect of the refueling boom remains always unfavorable in the range of angle of attack considered.

5. Results of the hover mode simulation, which agree fairly well with measurements, may provide useful data in assessing the potential hazards of the outflow flowfield.

6. The generally good agreement between the present CFD results and flight test data suggests that the present CFD model may serve as a practical tool in predicting the aircraft aerodynamic performance at a cost well within the available computer resources.

Acknowledgment

The present work was supported by the Naval Air Systems Command (Airtask A511-5115/010D/3H14250000) under the cognizance of S.A. Woods. The DoD HPC facilities at NAVO, CEWES, and WPAFB, as well as the NASA Ames Research Center provided the Cray C-90 CPU time. The author wishes to thank S.A. Woods and D. Grove of NAVAIR, J. Vorwald of NSWC-Carderock, M.A. McVeigh of Boeing Defense & Space Group, T. Wood of Bell Helicopter Textron, T.L. Holst of NASA Ames, and C. Rumsey of NASA Langley for useful discussions and comments during the course of the present work.

References

- ¹ McVeigh, M.A., Grauer, W.K., and Paisley, D.J., "Rotor Airframe Interactions on Tiltrotor Aircraft," *Journal of American Helicopter Society*, Vol. 35, No. 3, July 1990.
- ² Blohm, R. and Bartie, K., "V-22 Aerodynamic Report," Rept No. 901-909-632, Bell-Boeing Tiltrotor Team, July 1986.
- ³ Felker, F.F., Shinoda, P.R., Hefferman, R.M. and Sheehy, H.F., "Wing Force and Surface Pressure Data from a Hover Test of a 0.658-Scale V-22 Rotor and Wing," NASA Technical Memorandum 102244, Feb 1990.
- ⁴ Meyerhoff, C. and Gordge, D., "Navy Development Test (DT-IIA) of the MV-22 Aircraft, Contributory Test Report, Rotor Downwash," Technical Report, Naval Air Test Center, Patuxent River, MD, 1990.
- ⁵ Grauer, W.K. and Farrell, M.K., "V-22 Flight Test Aerodynamics," Technical paper presented at the 47th Annual Forum of American Helicopter Society, Phoenix, AZ, 1991.
- ⁶ Schleicher, D.R., and Alexander, H.R., "Technology Development for Advanced Tiltrotor Transports," AIAA Paper 93-4810, Dec 1993.
- ⁷ McVeigh, M.A., Liu, J., O'Toole, S.J., and Woods, S.A., "V-22 Osprey Aerodynamic Development -- A Progress Review," Paper No. 2225 presented at 22nd European Rotorcraft Forum, Brighton, UK, Sept 1996; also *Journal of Royal Aeronautical Society*, June-July, 1997.
- ⁸ Tai, T.C. and Vorwald, J., "Simulation of V-22 Rotorcraft Hover Flowfield," AIAA Paper 93-4878, 1993 AIAA International Powered Lift Conference, Santa Clara, CA, Dec 1993.
- ⁹ Tai, T.C., "Simulation and Analysis of V-22 Tiltrotor Aircraft Forward-Flight Flowfield," AIAA Paper 95-0045, 33rd AIAA Aerospace Science Meeting & Exhibit, Reno, NV, Jan 1995; also *Journal of Aircraft*, Vol. 33, No. 2, March-April 1996. pp. 369-376.
- ¹⁰ Tai, T.C., "Simulation and Analysis of V-22 Aircraft Maneuvering Aerodynamics," AIAA Paper 96-2477, 14th AIAA Applied Aerodynamics Conference, New Orleans, LA, June 1996.
- ¹¹ Tai, T.C., "Extension of Baldwin-Lomax Turbulence Model to Three-Dimensional Flows," AIAA Paper 97-0209, 35th AIAA Aerospace Sciences Meeting & Exhibit, Reno, NV, Jan 1997.
- ¹² Tai, T.C., "Effect of External Components on V-22 Aircraft Forward-Flight Aerodynamics," AIAA Paper 99-0532, 37th AIAA Aerospace Sciences Meeting & Exhibit, Reno, NV, Jan 1999.
- ¹³ Meakin, R., "Moving Body Overset Grid Methods for Complete Aircraft Tiltrotor Simulations," AIAA Paper 93-3350, July 1993.
- ¹⁴ Sorenson, R.L., "The 3DGRAPE Book: Theory, Users' Manual, Examples," NASA Technical Memorandum 102224, July 1989.
- ¹⁵ Flores, J., and Holst, T.L., "Numerical Solution of the Navier-Stokes Equations for Complex Configurations," *Lecture Series in Computational Fluid Dynamics*, Univ. of Tennessee Space Inst., Tullahoma, TN, March 1988.
- ¹⁶ Thomas, J.L., Krist, S.T., and Anderson, W.K., : "Navier-Stokes Computations of Vortical Flows Over Low-Aspect-Ratio Wings," *AIAA Journal* Vol. 28, No. 2, Feb 1990. pp. 205- 212.
- ¹⁷ Baldwin, B. and Lomax, H., "Thin Layer Approximation and Algebraic Model for Separated Turbulent Flow," AIAA Paper 78-0257, Jan 1978.
- ¹⁸ Spalart, P. and Allmaras, S., "A One-Equation Turbulence Model for Aerodynamic Flows," AIAA Paper 92-0439, Jan 1992.
- ¹⁹ Rumsey, C.L. and Vatsa, V.N., "A Comparison of the Predictive Capabilities of Several Turbulence Model Using Upwind and Central-Difference Computer Codes," AIAA Paper 93-0192, Jan 1993.

- ²⁰ Degani, D. and Schiff, LB, "Computation of Supersonic Viscous Flows Around Pointed Bodies at Large Incidence," AIAA Paper 83-0034, Jan 1983.
- ²¹ Tai, T.C., "Theoretical Calculation of Viscous-Inviscid Transonic Flows," *Lecture Series on Shock-Boundary Layer Interaction in Turbomachines*, von Karman Institute for Fluid Dynamics, Brussels, Belgium, Jun 1980.
- ²² McVeigh, M.A., Liu, J., and Wood, T., "Aerodynamic Development of a Forebody Strake for the V-22 Osprey," AHS 51st Annual Forum, May 1995.
- ²³ Visbal, M.R., "Onset of Vortex Breakdown Above a Pitching Delta Wing," *AIAA Journal*, Vol. 32, No. 8, Aug 1994. pp. 1568-1575.
- ²⁴ Tai, T.C. and Walker, M., "On Three-Dimensional Flow Separation Criteria," AIAA Paper 91-1740, Jun 1991.
- ²⁵ Delery, J.M., "Physics of Vortical Flows," *Journal of Aircraft*, Vol. 29, No. 5, Sept-Oct 1992, pp. 856-876.
- ²⁶ Johnson, W., "A Comprehensive Analytical Model of Rotorcraft Aerodynamics and Dynamics," Johnson Aeronautics, Palo Alto, CA, 1988.
- ²⁷ Meyerhoff, C., and Gorge, D., "Navy Development Test (DT-IIA) of the MV-22 Aircraft, Contributory Test Report, Rotor Downwash," Technical Report, Naval Air Test Center, Patuxent River, MD, 1990.